Route Constraint Editor user manual

(Version 2.0)

Shenzhen Unisplendour Tongchuang Electronics Co., Ltd.

Document version revision record

VERSION NUMBER	RELEASE DATE	REVISION HISTORY
V1.0	2022.07.12	initial version
V2.0	2022.12.29	Adding Floor Plan window description; design browser content integration. Add clock region display description; update reload window form; new Add tile layer density display and special resource coloring.

Table of contents

1. FEATURES 5
2. INTRODUCTION TO HOW TO OPEN RCE 6
2.1 OPEN RCE BY CLICKING THE TOOLBAR ICON OF THE PDS INTERFACE 6
2.2 OPEN RCE BY CLICKING THE ROUTE CONSTRAINT EDITOR OPTION UNDER TOOLS IN THE MENU BAR..6
3. TOOLBAR AND MENU INTRODUCTION 9
3.1 TOOLBAR 9
3.1.1 File operation and window start button 9
3.1.2 View series buttons 9
3.1.3 Other buttons 10
3.2 RIGHT-CLICK MENU 10
3.2.1 Toolbar right-click menu 10
3.2.2 Device window right-click menu 11
3.2.3 Other right-click menus 12
4. WINDOW INTRODUCTION 16
4.1 DEVICE VIEW WINDOW 16
4.2 FLOOR PLAN WINDOW 16
4.3 DESIGN BROWSER WINDOW 17
4.4 CONSOLE WINDOW 19
4.5 SETTING LAYER WINDOW 19
4.6 BIRD'S EYE VIEW WINDOW 21
5. DETAILED OPERATION 23
5.1 MANUALLY SPECIFY THE WIRING PATH 23
5.1.1 Wiring 23
5.1.2 Disconnecting stitches 27
5.2 ASSIGN NET PRIORITY ROUTING 28
5.3 SPECIFY INST PRIORITY ROUTING 28
5.4 CANCEL NET PRIORITY WIRING 28
5.5 CANCEL INST PRIORITY ROUTING 29
5.6 FILE CONSTRAINTS 29
5.7 MARK FUNCTION 29
5.8 OTHER FUNCTIONS 31
5.9 INPUT COMMANDS 32
5.9.1 Close 32
5.9.2 get_path_delay 32
5.9.3 Help 33
5.9.4 manual_route 33
5.9.5 next 33
5.9.6 save_adif 34
5.9.7 select 34
5.9.8 show 35
5.9.9 unroute 35
6. HOW TO MANUALLY SPECIFY THE WIRING PATH DIRECTLY USING THE RCF FILE 37
7. CONTACTS 39

Figure catalog

Figure 1-1 Route Constraint Editor interface 5
Figure 2-9 Route Constraint Editor under Tools 6
Figure 2-10 Input Route Constraint File without RCF file in PDS 7
Figure 2-11 Input Route Constraint File with RCF file in PDS 7
Figure 2-12 RCE window with PDS Place&Route in out of date state....8
Figure 3-1 File operation and window start button 9
Figure 3-2 view series button....9
Figure 3-3 other buttons....10
Figure 3-4 Toolbar right-click menu....10
Figure 3-5 Device Window right-click menu....11
Figure 3-6 The right-click menu of inst....13
Figure 3-7 The option option pop-up window of the generate constraint function....14
Figure 4-1 Device window 16
Figure 4-2 Floor Plan view window....17
Figure 4-3 Design Browser icon diagram....17
Figure 4-4 Schematic diagram of Design Browser Inst window....18
Figure 4-5 Design Browser window toolbar 18
Figure 4-6 Design Browser window....18
Figure 4-7 Console window 19
Figure 4-8 Setting window 20
Figure 4-9 Bird's eye view window....22
Figure 5-1 Selecting branches for manual routing 23
Figure 5-2 Select the first pin 24
Figure 5-3 Click the interface to select the pin....24
Figure 5-4 Manual Routing Window....24
Figure 5-5 Add the next pin....25
Figure 5-6 Replacing the pins....25
Figure 5-7 Click “Manual Route” after specifying the route....26
Figure 5-8 Manual wiring branch successful....26
Figure 5-9 Wiring failure prompt 27
Figure 5-10 Select a branch to remove stitches 27
Figure 5-11 Deleting a certain branch of net....28
Figure 5-12 Prior Route Nets....28
Figure 5-13 Prior Route Nets....28
Figure 5-14 UnPrior Route Nets....29
Figure 5-15 UnPrior Route Insts....29
Figure 5-16 Setting up Mark 30
Figure 5-17 Device after setting Mark....30
Figure 5-18 Mark browser button....30
Figure 5-19 Mark browser window 31
Figure 5-20 Local zoom view....32
Figure 5-21 Command Run Scrollbar....32
Figure 5-22 background scroll bar....32
Figure 6-1 Show -design_net nt_GTP_LUT4_Z Execution Result....37

1. FEATURES

This document describes in detail the various functions and usage methods of Route Constraint Editor (hereinafter referred to as RC E). Many functions of Route Constraint Editor are similar to Design Editor MPNR mode), but the RCE interface strictly prohibits layout operations, otherwise the resulting layout constraint file (RCF) is likely to be invalid.

In RCE, users can make 3 kinds of routing constraints. Once the constraints are completed, they can be saved to the specified constraint file. Then when the user runs Place&Route in Flow, the routing operation will be completed according to the user routing constraints.

The Route Constraint Editor interface is shown in Figure 11:

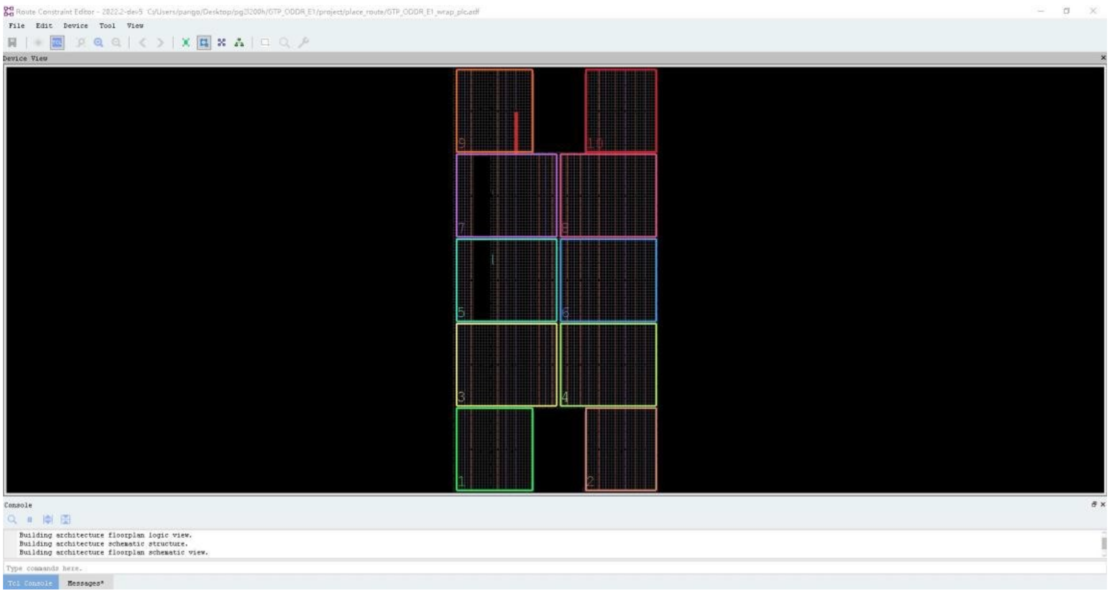


Figure 1-1 Route Constraint Editor interface

The functions and operation steps of RCE will be introduced in detail below..

2. INTRODUCTION TO HOW TO OPEN RCE

Open Pango Design Suite (hereinafter referred to as PDS software) and create a new project; after the new project is completed, RCE can be started in the following ways.

2.1 Open RCE by clicking the toolbar icon of the PDS interface

Since the place DB (suffix _plc.adf) file needs to be loaded when RCE is started, to open RCE in the PDS software, the Device Map needs to be run in Flow, and then the layout will be completed during the process of starting RCE to generate the place DB and its Load to open the RCE interface.

Click the icon shown in the figure in the toolbar of the PDS interface. If dev_map and subsequent actions have been run, the RCE startup interface shown in Figure 2-6 will be opened immediately; if dev_map has not been run, it will wait until dev_map After running, open the RCE startup interface as shown in Figure 2-7.

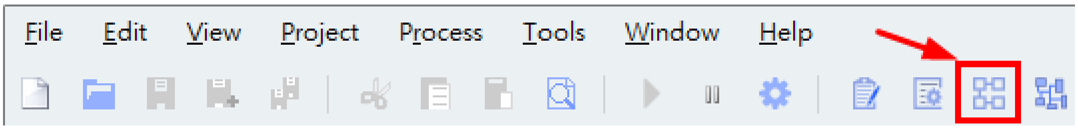


Figure 2-8 RCE startup icon in the toolbar

2.2 Open RCE by clicking the Route Constraint Editor option under Tools in the menu bar

Another way to open RCE in the PDS software is to click the Route Constraint Editor option under Tools in the menu bar, as shown in the figure.

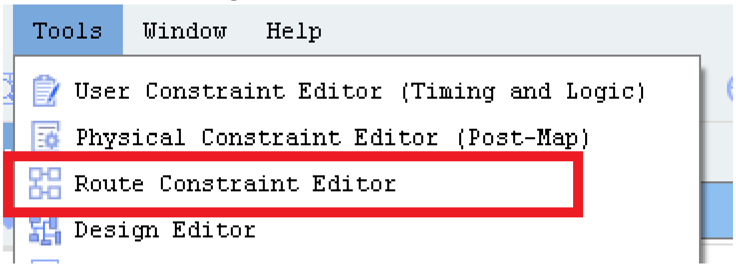


Figure 2-9 Route Constraint Editor under Tools

After clicking, it is necessary to open the startup interface as shown in the figure below after dev_map runs.

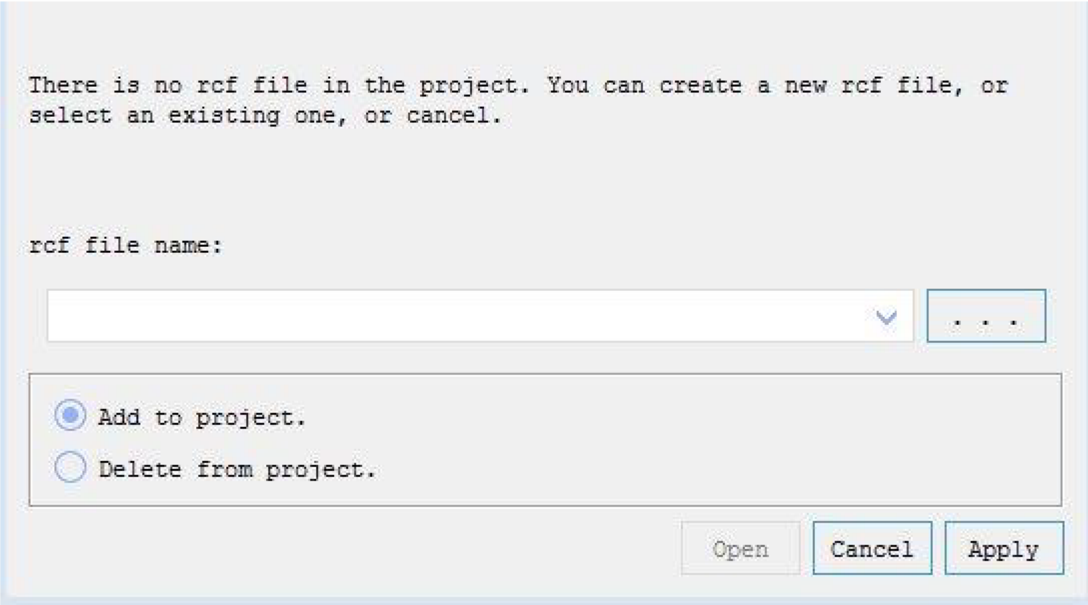


Figure 2-10 Input Route Constraint File without RCF file in PDS

When starting RCE in PDS, if there is no Route Constraint File (hereinafter referred to as RCF) file in the original PDS project, in the pop-up dialog box of importing RCF files, the content of the text box is empty, as shown in the figure below.

When there is an RCF file in the original PDS project, the content of the text box in the pop-up input RCF file dialog box is the existing RCF file in the project, as shown in the figure.

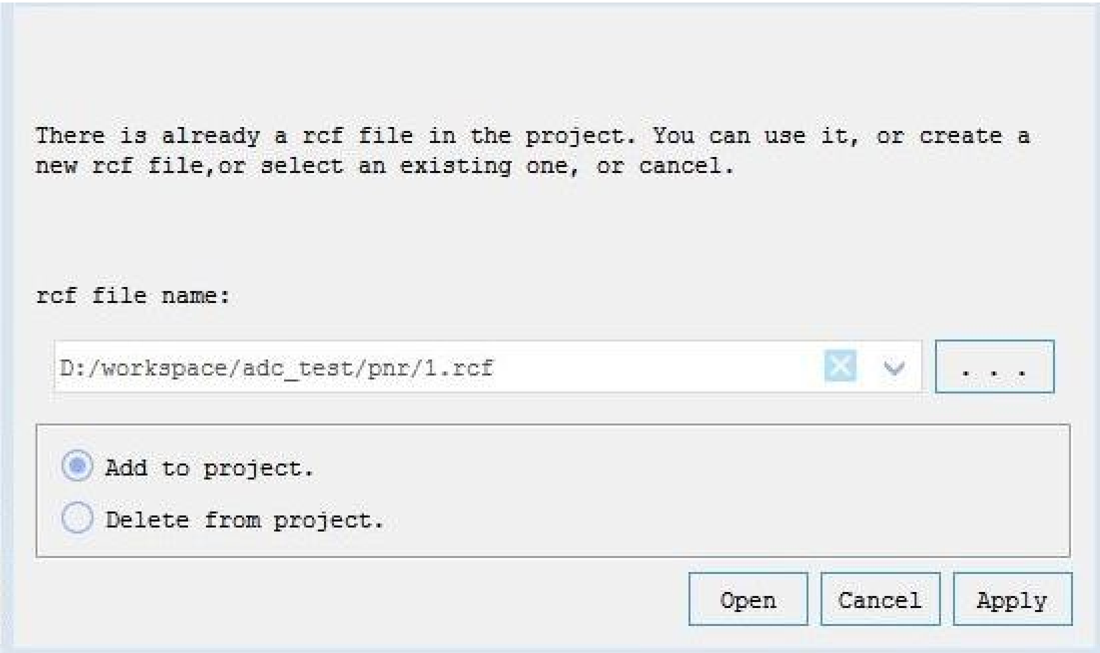


Figure 2-11 Input Route Constraint File with RCF file in PDS

When the Place&Route in the PDS project is in the out of date state, the pop-up input RCF file dialog box when entering RCE will add the option Use Device Map DB. After checking, it will use mapDB to enter RCE, otherwise it will load the PLC DB in the out of date state file, as shown in the figure.

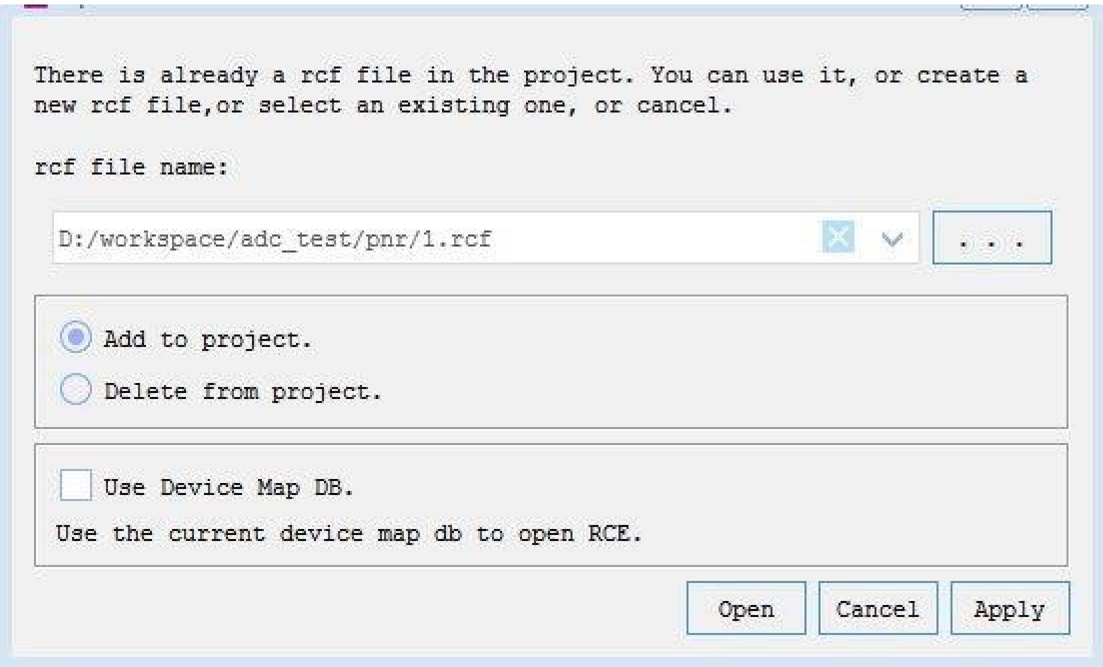


Figure 2-12 RCE window with PDS Place&Route in out of date state

[Add to project.]: You can directly input the name of the RCF file to be created or select an existing RCF file to add to the PDS project, then click the [Open] button to load the RCF file into RCE. If the input is a legal RCF file name that does not exist, an RCF file with this name will be created in the working directory after clicking the [Open] button (you can manually add the suffix .rcf or no suffix, if no suffix Then the software will automatically add the suffix .rcf when creating a new RCF file). If the selected RCF is inconsistent with the original RCF file in the project, the selected RCF file will replace the original RCF file in the project and affect the status of the PDS project. If the text box is empty, the [Open] button will not be available (as shown in the figure).

[Delete from project.]: After selection, click the [Apply] button to remove the existing RCF file in the PDS project, and RCE will not be opened.

3. TOOLBAR AND MENU INTRODUCTION

The following describes the purpose of the toolbar buttons and all menus in detail.

3.1 Toolbar

3.1.1 File operation and window start button

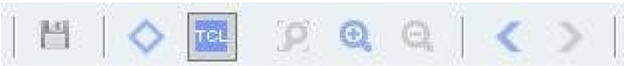


Figure 3-1 File operation and window start button

【Save】: Save the DB file.

【Mark Browser】: Open or close the Mark Browser window.

【Console】: Open or close the Console window.

【View All】: Display global view.

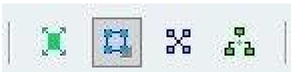
【Zoom In】: Enlarge view.

【Zoom Out】: Zoom out the view.

【Previous View】: previous view.

【Next View】: Next view.

3.1.2 View series buttons


Figure 3-2 view series button

【Show Place Resource Only】: Device View/Floor Plan view switching

【Show Placement】: Show/hide the layout result.

【Show Routes】: Show/hide routing results.

【Show Airline】: Show/hide the flight line.

【Show Rats】: Show/hide the minimum spanning tree of unrouted net.

(Show Routes, Show Airline, and Show Rats depend on the Show Placement function, and these three items can only be selected on the basis of selecting Show Placement)

3.1.3 Other buttons



Figure 3-3 other buttons

【Show Report Region】: Display the report and flying lines in the region.

【Search Device Inst】: Open the search box to search for device instance, pin and package pin. Pin search methods, for example: Device/pin (slash), Device.pin (dot), Device:pin (colon, pin after), pin:Device (colon, pin before), the search supports fuzzy matching, The search results display the search result list, the up and down keys switch the selected content in the list, and the enter key and direct click can locate.

【Setting】: Open the Setting interface to set the Device to display relevant information, including color and other settings,

3.2 Right-click menu

The purpose of all right-click menu options is detailed below.

3.2.1 Toolbar right-click menu

Right-click the toolbar, and the right-click menu will appear, as shown in Figure 3-4.

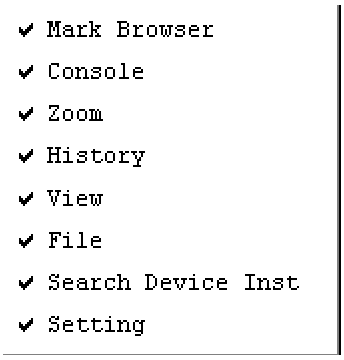


Figure 3-4 Toolbar right-click menu

【Mark Browser】: Show or hide the Mark Browser window

【Console】: Show or hide the Console window.

【Zoom】: Show or hide the Zoom series buttons on the toolbar.

【History】: Show or hide the History series buttons on the toolbar.

【View】: Show or hide the View series buttons on the toolbar.

【File】: Show or hide the buttons related to file operation in the toolbar.

【Search Device Inst】: Show or hide the Search Device Inst button.

【Setting】: Show or hide the Setting button.

3.2.2 Device window right-click menu

If you want to view the usage of various routing resources in the routing results, click the Show Routes button on the toolbar, then right-click in the Device window, and the menu shown in Figure 3-5 will appear

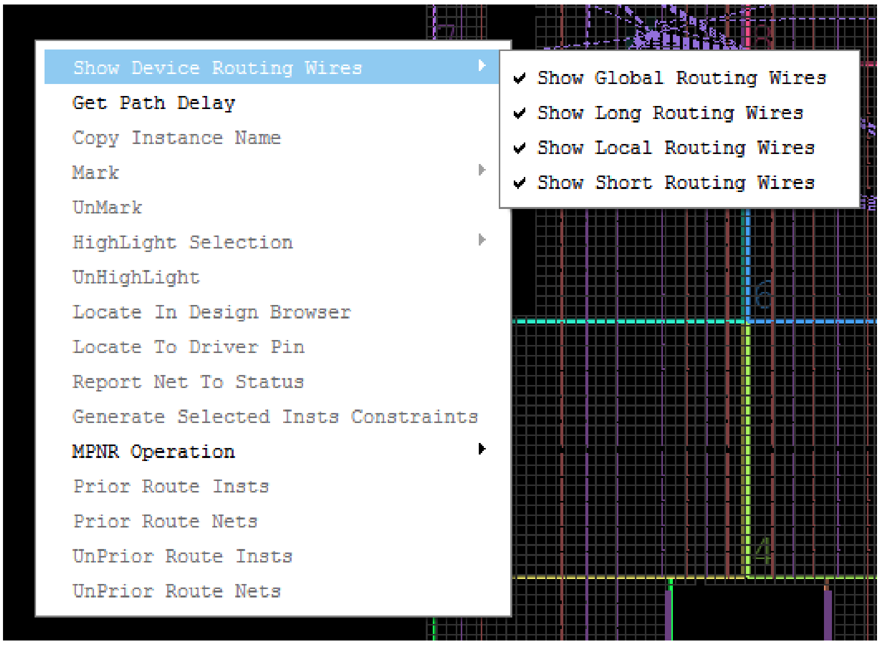


Figure 3-5 Device Window right-click menu

【Show Device Routing Wires】: Put the mouse on this option, different series of devices will display the following different menu items.

【Show Local Routing Wires】: Display the Local wires in the routing.

【Show Global Routing Wires】: Display Global Routing Wires.

【Show Long Routing Wires】: Show Long Routing Wires in routing.

【Show Short Routing Wires】: Show Short Routing Wires.

【Show Logical Routing Wires】: Display Logical Routing Wires.

【Show Clock Routing Wires】: Display the clock wires in the routing.

【Show Quarter Routing Wires】: Display Quarter wires in routing.

【Show Double Routing Wires】: Show Double Routing Wires in routing.

【Show Logical Routing Wires】: Display Logical Routing Wires.

Different series support different types of wire, you can check the specific support in the relevant position in the DE user manual (in chapter 3.2.2)

【Get Path Delay】: Input and calculate parameters of Get Path Delay

【Copy Instance Name】: Copy the instance name.

【Mark】: mark function, you can mark the selected object.

【Unmark】: Cancel the mark function, cancel the mark of the mark object that has been set

【HighLight Selection】: Support highlighting and modify the color of the selected net or inst.

【UnHighLight】: Change the selected and modified net or inst back to the default color.

【Locate To Design Browser】: select and jump to the corresponding location of Design Browser

【Locate To Driver Pin】: Select and jump to the driver pin position of net

【Report Net To Status】: Print the routing path of the selected net

【Generate Selected Insts Constrains】: Open the generate constraint function window

【MPNR Operation】contains the following two sub-options:

【UnRoute】: Uncheck the routing of instance or net.

【Manual Route】: Manually route the selected net.

【Prior Route Insts】: After the inst is selected, it will be marked, and the marked inst will give priority to routing the net passing through the inst in the pnr.

【Prior Route Nets】: After the net is selected, it will be marked, and the marked net will give priority to the routing of the net in the pnr.

【UnPrior Route Insts】: This option can only be clicked after the marked inst is selected, and the mark of the inst will be canceled after clicking.

【UnPrior Route Nets】: This option can only be clicked after the marked net is selected, and the mark of the net will be canceled after clicking.

3.2.3 Other right-click menus

Right-click the Design Browser button, select the Design button, and click instance in the window to pop up the right-click operation menu, as shown in Figure 3-6.

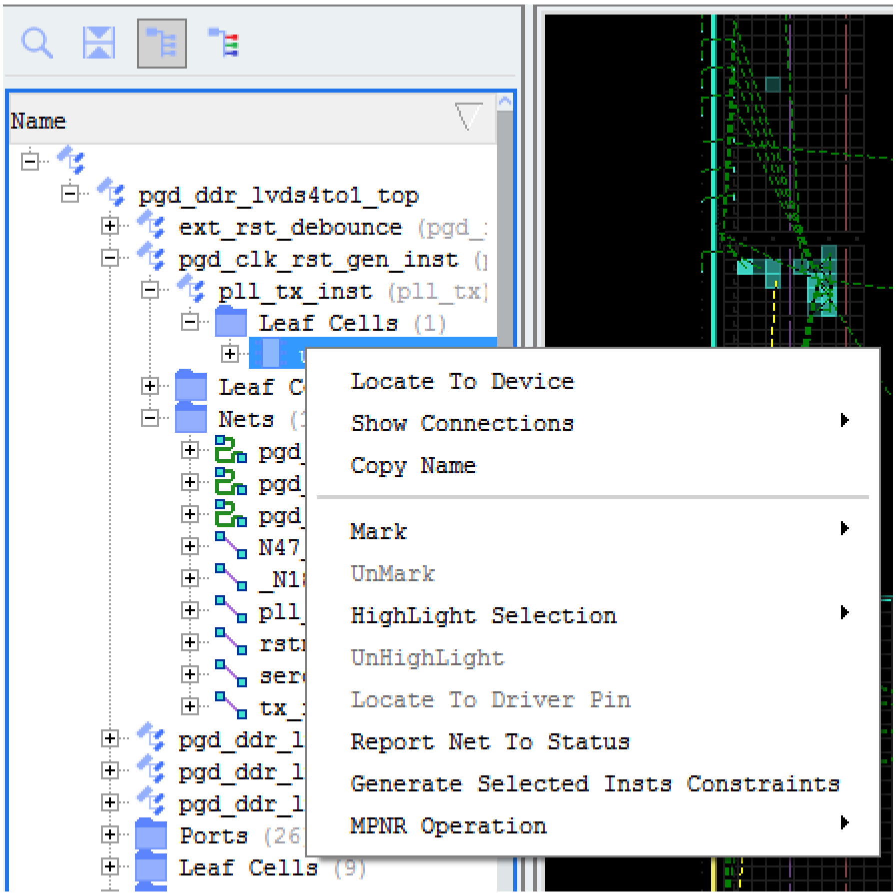


Figure 3-6 The right-click menu of inst

【Locate To Device】: Display the selected part in Device.

【Show Connections】: Display the inst connection relationship menu of the selected level

【Show Internal Connections】: Display the connection relationship between insts of the selected level.

【Show External Connections】: Display the insts of the selected level and the external connections.

【Show All Connections】: Display all connections of inst at the selected level.

【Copy Name】: Copy the name of the selected item.

【Mark】: Mark the position of the item in the device view.

【UnMark】: Unmark item in device view

【Highlight】: The item selected by the highlight will recurse to the net item.

【UnHighlight】: The item selected by unhighlight will recurse to the net item.

【Locate To Driver Pin】: Select and jump to the driver pin position of net

【Report Net To Status】: Print the routing path of the selected net

【MPNR Operation】: Same as the menu in Device view, the object selected in Design Browser performs MPNR operation.

【Generate Selected Insts Constraints】: Click to select the right-click menu of inst and hierarchy,

and the option Generate Selected Insts Constrains is highlighted and optional. Users can click this option to print the selected inst constraints information to the specified file. Users can configure the required Constraint options, each option has tips instructions, users can also open the page through Save Constrain to pcf in the file item in the menu bar, as shown in the figure below.

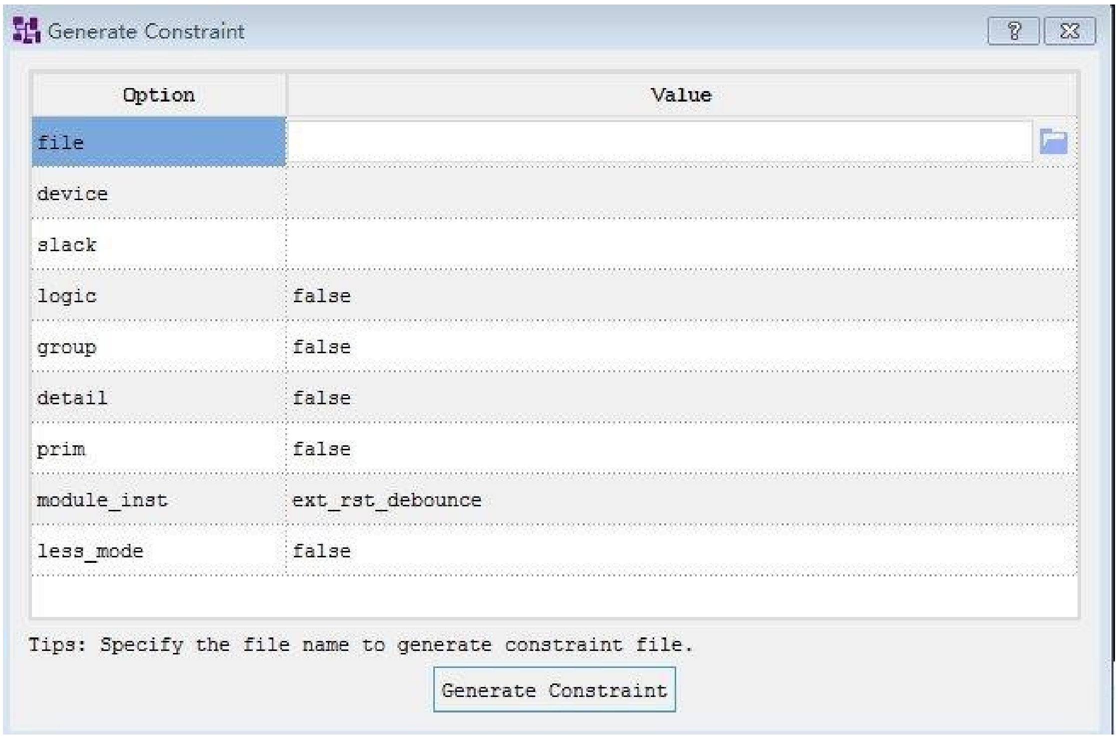


Figure 3-7 The option option pop-up window of the generate constraint function

Each option annotation:

File: This option must be given. And at present, the content of the specified file will be cleared every time, and then the new content will be rewritten. Therefore, you need to pay attention to backup the original content in the specified file.

Device: This option mainly prints the constraints of the specified device

Logic: This option is to generate the selected level constraint information in the form of fdc

Group: This option is to generate group instance constraint information in the form of pcf

Detail: This option is used together with the group to print the constraint information of each inst in the group

Prim: This option is to generate the constraint information of the prim instance in the form of pcf

Module_inst: This option is to print the position constraints of design inst starting with the specified string according to the user's needs (the top level can be manually modified, and the item of other levels is the name of the level and cannot be modified)

Slack: This option is to print the location constraint of inst whose slack value is greater than or equal to the number given by the user

Less_mode: This option is used with slack, and adding -less_mode can print the position constraint of inst with slack value less than number

Only_cons: When only_cons is true, the exported data only includes the location information of the design instance that is constrained in the constraint file or applied as a position constraint in DE

4. WINDOW INTRODUCTION

The functions of each window and related operation methods are introduced in detail below.

4.1 Device View window

This window can display chip structure, all resources, layout position, wiring path, occupied resources, and can perform operations such as manual wiring, as shown in Figure 4-1.

Mouse operation on the Device View interface:

Mouse wheel: Swipe up and down on the Device View interface.

CTRL+mouse wheel: Zoom in and out on the Device View interface.

SHIFT+mouse wheel: Control the Device View interface to slide left and right.

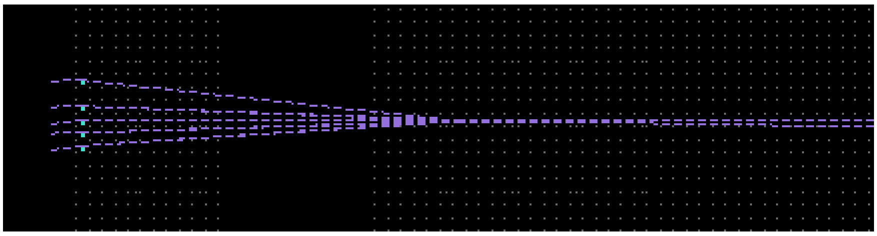


Figure 4-1 Device window

4.2 Floor Plan window

Click the Show Place Resource Only button, the interface will switch to display the layout resources, not the status of the routing resources, and most of the layout resources will be enlarged, and the routing can still be displayed, but in a form similar to the flying wire, as shown in the figure below. Click again to switch the FloorPlan view back to the Device view.

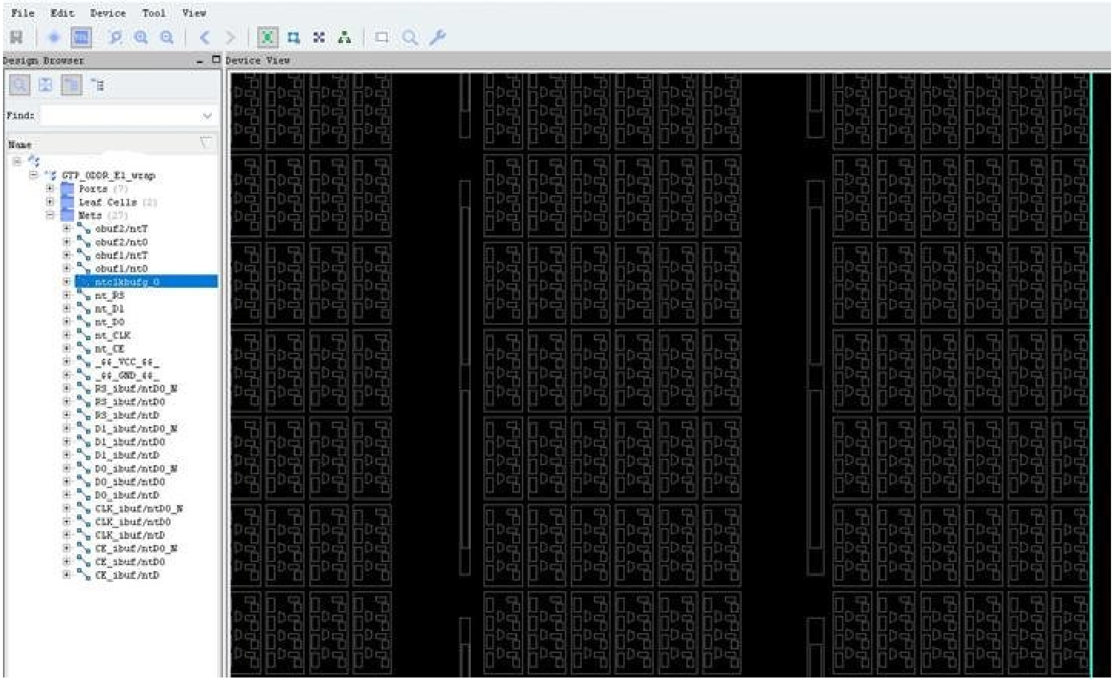


Figure 4-2 Floor Plan view window

4.3 Design Browser window

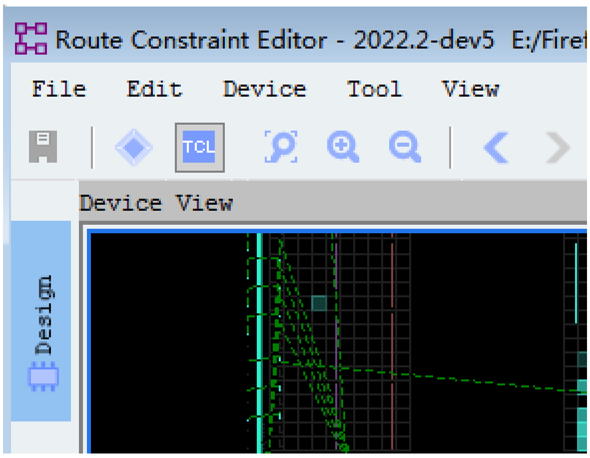


Figure 4-3 Design Browser icon diagram

Browser currently exists in the left tab bar, select to open browser, and the currently selected Design Browser window will be displayed. This window can search for instance and net, as shown in the figure below.

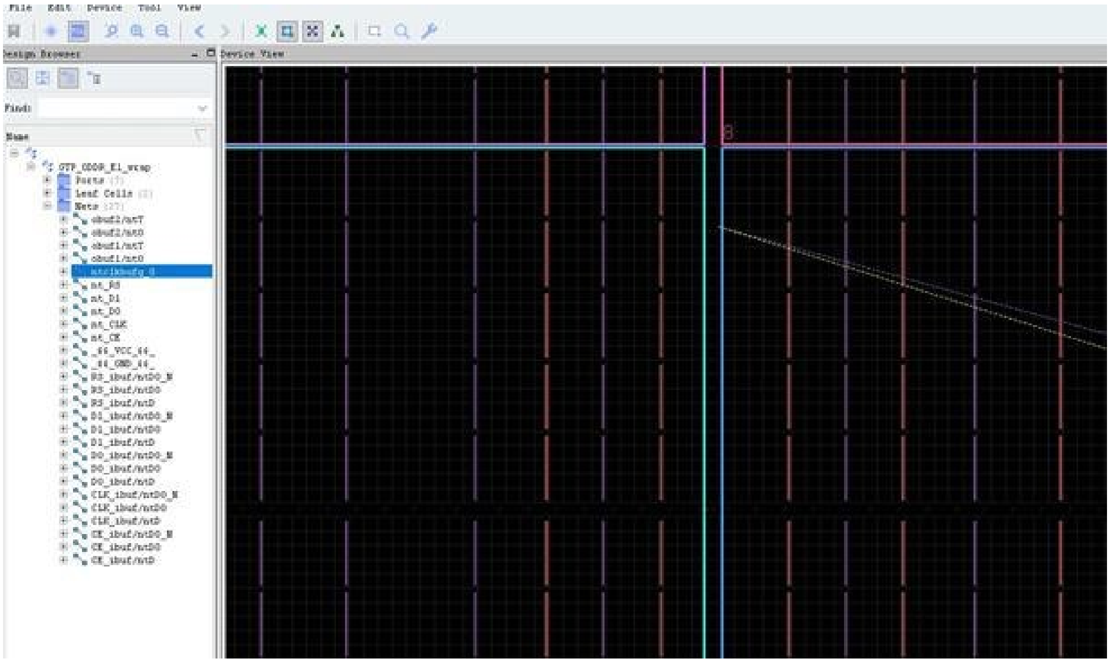


Figure 4-4 Schematic diagram of Design Browser Inst window

In the opened window, there is a toolbar on the upper side, as shown in the figure below:

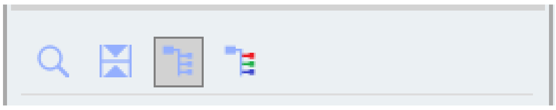


Figure 4-5 Design Browser window toolbar

From left to right are: Search, Collapse All, Switch Hierarchy Structure, Switch Only Display Highlight.

In the Design window, the top-level node displays device-related information, the top-level node displays the design netlist name information node, the hierarchy node and port node are placed under the design node, and the hierarchy contains CarryChain, Group, Leaf Cell, Net and other nodes, CarryChain, Group, Leaf Cell contains instance nodes, instance contains pin nodes, and Net contains net bus and net nodes.

In the initial state, the title is only Name, and the following items can be hidden and displayed by right-clicking on it to open the right-click menu, mainly including Site (displaying the instance layout position), Fanout and Bounding Box, and resource count, as shown in the figure below.

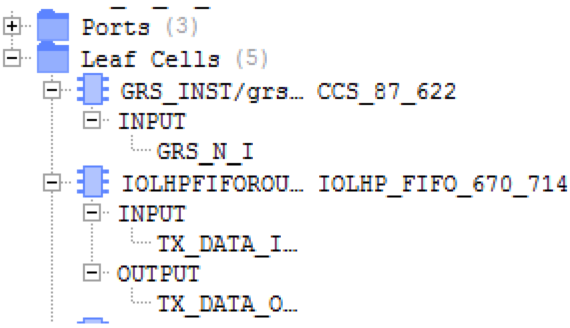


Figure 4-6 Design Browser window

Switch to show only the Highlight button (only show highlight). After clicking, the highlighted object can be filtered out and displayed in the browser list. When there is no highlighted object, the browser list will be empty after clicking.

4.4 Console window

This window can execute wiring commands and display error warning information. Selecting device, instance, net, etc. can display its name, layout location and other related information, and can perform query and stop printing information functions, as shown in the figure below..



Figure 4-7 Console window

4.5 Setting Layer window

This window controls the display of resources in the device view interface, as shown in Figure 4. The line type, I/O Banks and Clock Region are different for different devices.

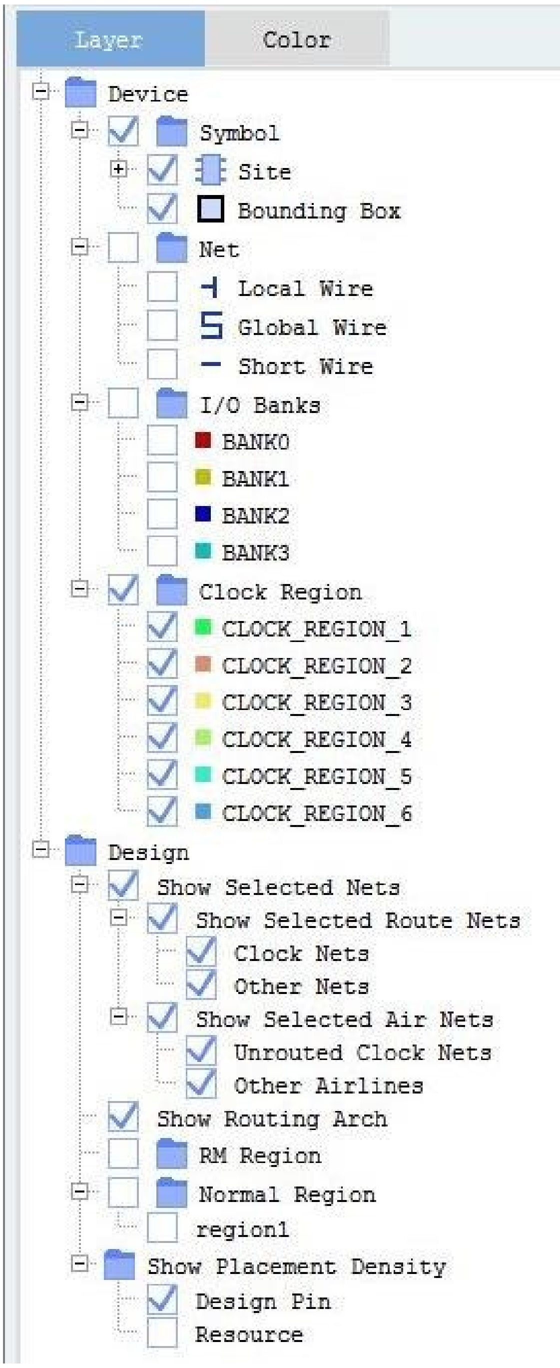


Figure 4-8 Setting window

【Site】: Show/hide the specified device resources.

【Bounding Box】: Show/hide device resource frame.

【Long Wire】: Show/Hide Long Wires.

【Short Wire】: Show/Hide Short Wires.

【Global Wire】: Show/Hide Global Wires.

【Local Wire】: Show/hide Local Wires.

Different series support different types of wire, you can check the specific support in the relevant position of the DE user manual (in chapter 3.2.2)

【I/O Bank】: Show/hide the specified IO Bank.

【Clock Region】: Show/hide the specified Clock Region.

【Show Selected Nets】: Show/hide the nets connected with instance.

【Show Selected Route Nets】: Show/hide the selected routing nets connected with instance.

[Clock Nets]: Show/hide the selected clock line connected to the instance; when this option is selected, the clock line will not be highlighted after the inst containing the clock line is selected on the device interface.

[Other]: Show/hide the selected non-clock wires connected to the instance; when this option is selected, after selecting the inst that contains the clock wire on the device interface, only the clock wire will be highlighted, and other wires will not be highlighted.

【Show Selected Air Nets】: Show/hide the selected air nets connected with instance.

[Unroute Clock Nets]: Show/hide the selected clock flying wires connected to the instance; when this option is selected, after selecting the inst containing the clock flying wires on the device interface, the clock flying wires will not be highlighted.

[Other airlines]: Show/hide the selected non-clock flying lines connected to the instance; when this option is selected, after selecting the inst that contains the clock flying lines on the device interface, only the clock flying lines will be highlighted, and other flying lines will not highlight.

【Show Routing Arch】: Show/hide the internal Routing Arch line.

[RM region]: Show/hide the RM region defined in the constraint file (this item exists in the local dynamic reconfiguration project)

[nomal region]: Show/hide the region defined in the constraint file

【Show Placement Density】: When the zoom level is small, the interface will display the density of placement placement. There are two ways to switch between Resource (resource density) and Design pin (default) density

4.6 Bird's eye view window

This window can display the position of the chip where the device view interface is located and the selected design net in real time. The window can be opened or closed by clicking the button between the scroll bars or ctrl+v, dragging the border to zoom in and out of the window, bird's eye view Displayed in the lower right corner of the interface, as shown in the figure below

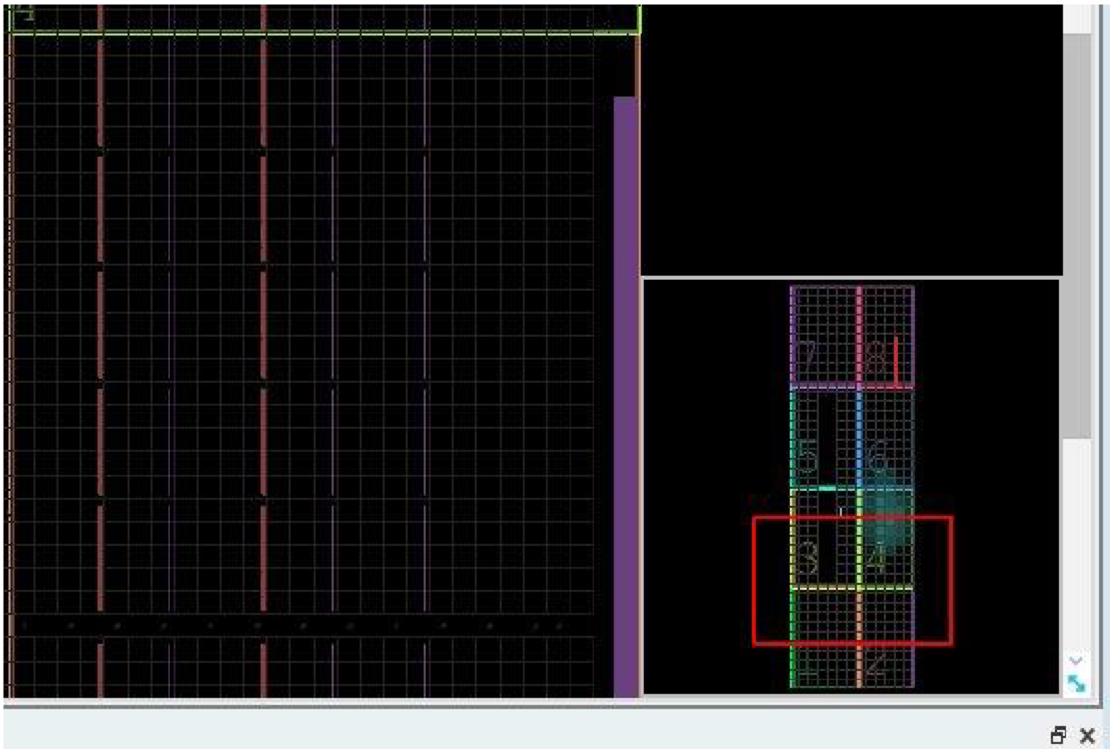


Figure 4-9 Bird's eye view window

5. DETAILED OPERATION

5.1 Manually specify the wiring path

5.1.1 Wiring

If you choose to manually specify the wiring path, you can only manually specify one branch path of the net at a time, and you must start from the starting point of the net, and specify the device pins passing through the wiring path step by step until you specify the end point on the net branch.

It supports manual routing of multiple branches of the same net, and after the routing is saved, a constraint command will be saved for each branch in the rcf file.

For the unwired net, the user can choose pins according to their own needs to form a route, and the route created by the user will be retained during wiring, and the rest will be automatically routed by the software.

This operation can only be performed on one branch of the net at a time, select the branch to be routed, and right-click to select "Manual Route", as shown in Figure 5-1. Note that the premise of using this function is that the instance or port where the driver pin and load pin of the net selected by the user have been constrained in the pcf file, otherwise even if the net is selected, Manual Route cannot be clicked.

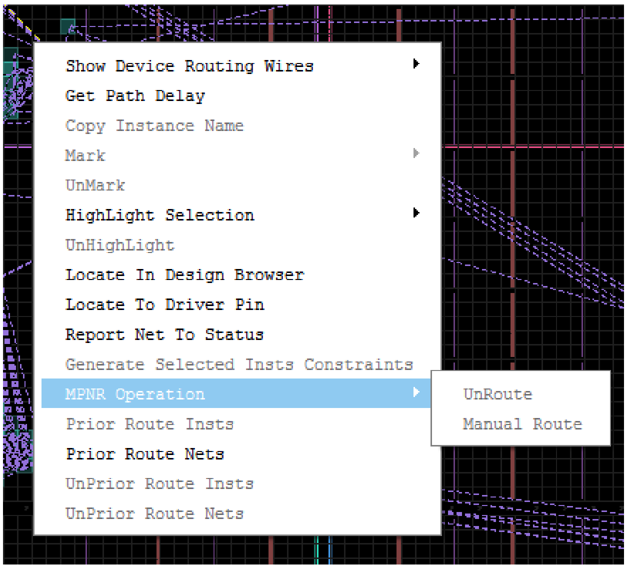


Figure 5-1 Selecting branches for manual routing

After clicking “Manual Route”, a selection window for the first pin will pop up. This window requires the user to select a pin as the first point of the route specified by the user. As shown in Figure 5-2.

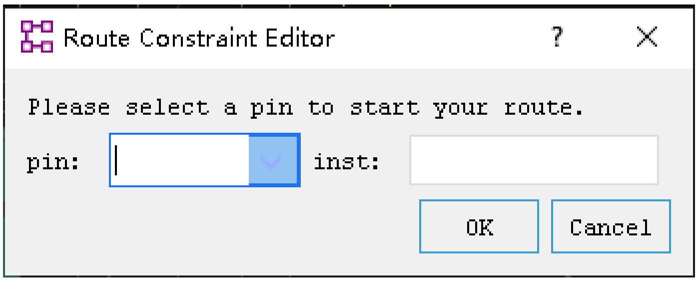


Figure 5-2 Select the first pin

The first pin can be selected by manually inputting the pin name and the name of the instance it belongs to, or directly clicking the pin icon on the interface with the mouse, and the pin information will be automatically read into the dialog box, as shown in the figure 5-3 shown.

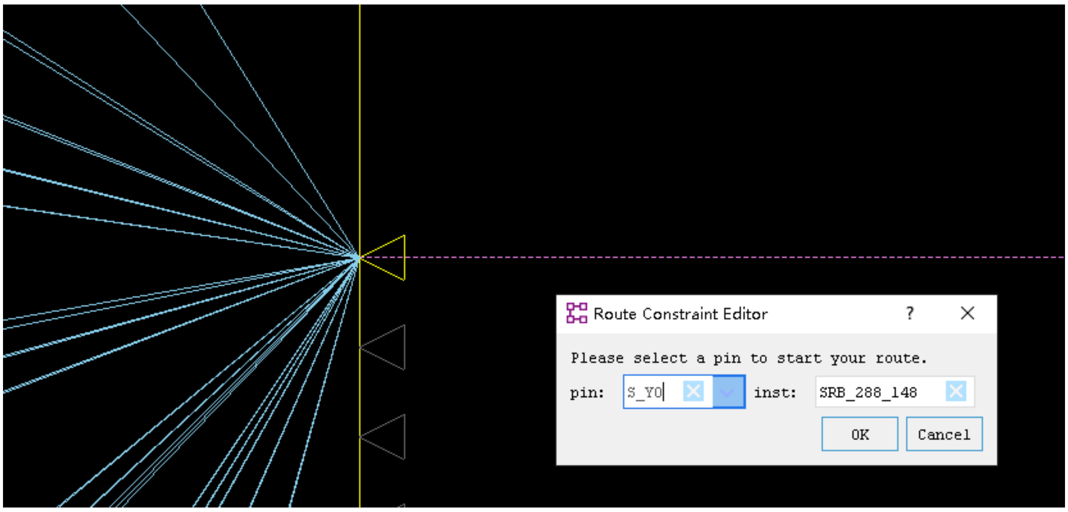


Figure 5-3 Click the interface to select the pin

Click "OK", and the manual wiring window will appear. From top to bottom, the current operating branch, net driver, and the first selected pin are displayed. At the same time, the path of the net in the interface will also change according to the selected pin, such as shown in Figure 5-4.

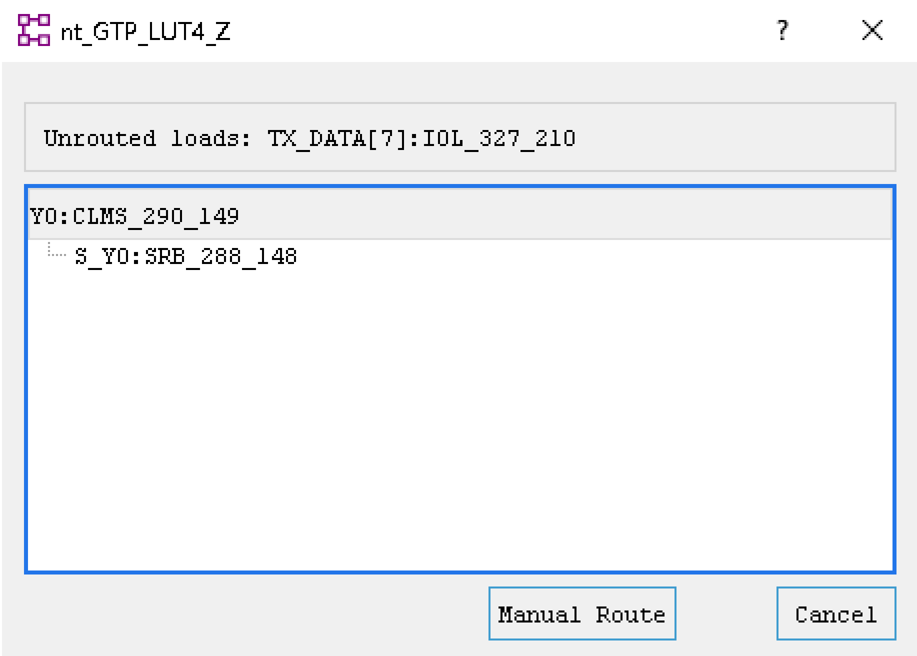


Figure 5-4 Manual Routing Window

The manually specified path consists of at least 2 pins. After selecting the first pin according to the above process, right-click the pin and select "Add Child Pin Item" to add the next pin, or "Delete Child Pin Item". Item" to delete the current pin, but cannot delete the first pin. As shown in Figure 5-5.

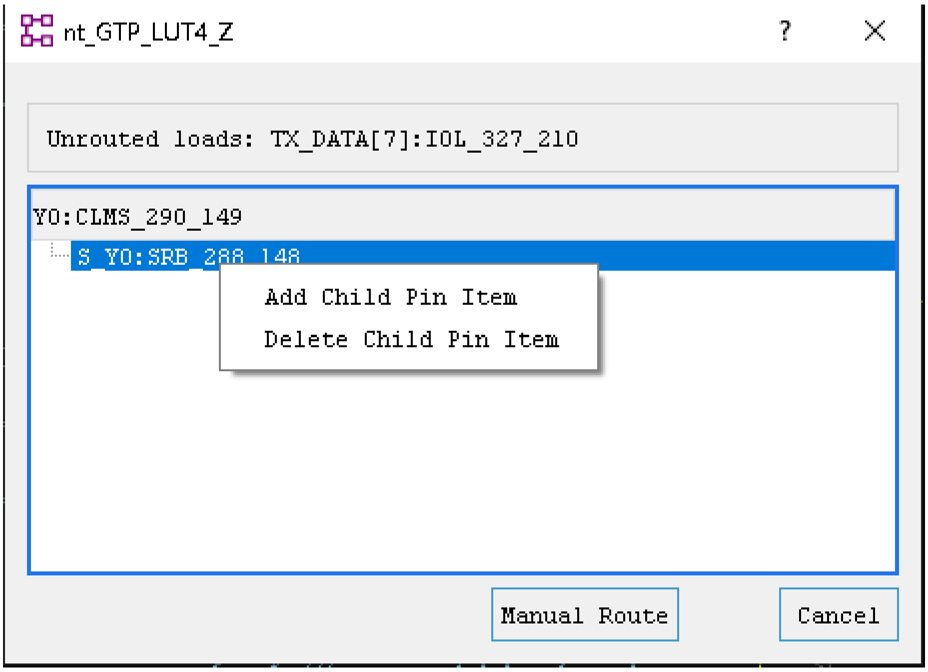


Figure 5-5 Add the next pin

"Add Child Pin Item" is automatically selected by the system for a pin connected to the previous pin. If there are multiple options, the system will select the first one by default, which can be changed by double-clicking the pin. As shown in Figure 5-6.

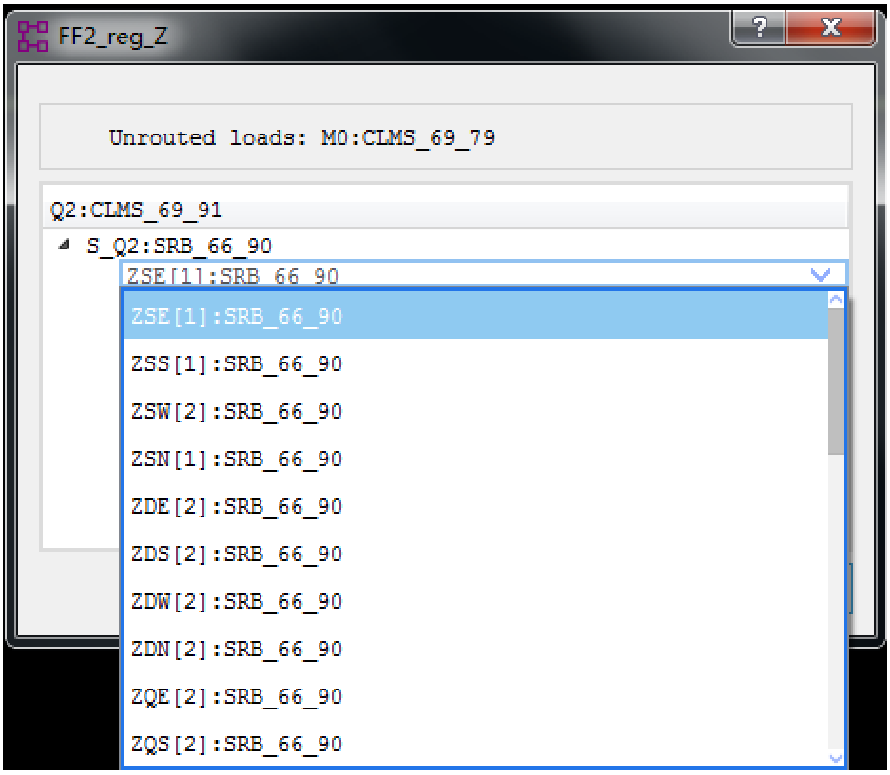


Figure 5-6 Replacing the pins

After selecting 2 or more pins, click "Manual Route" to complete the wiring of the branch, the

manually specified route is reserved, and the rest is automatically routed by the system, as shown in Figure 5-7.

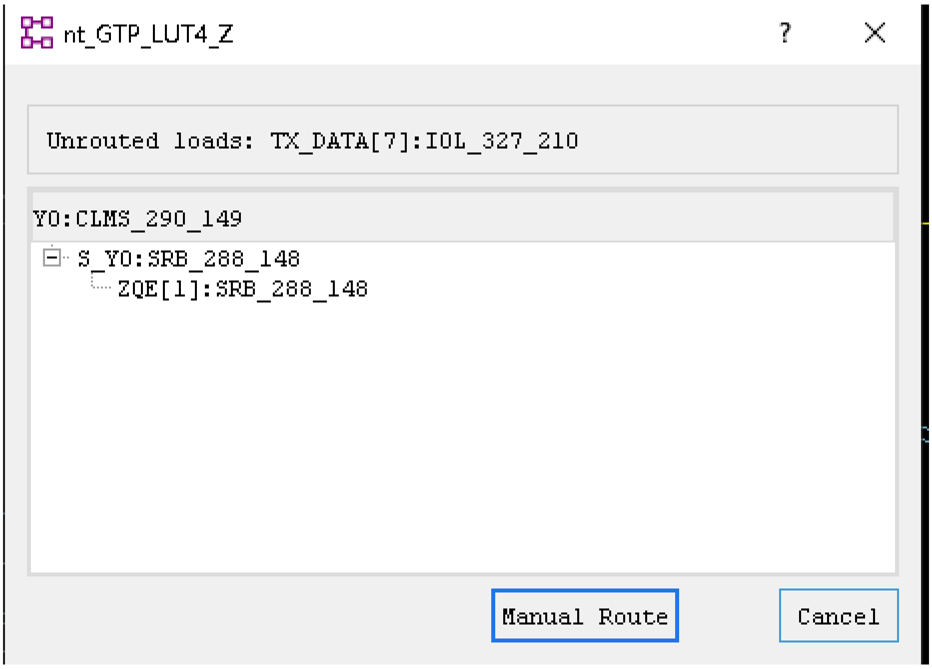


Figure 5-7 Click “Manual Route” after specifying the route

The wiring is successful, as shown in Figure 5-8.

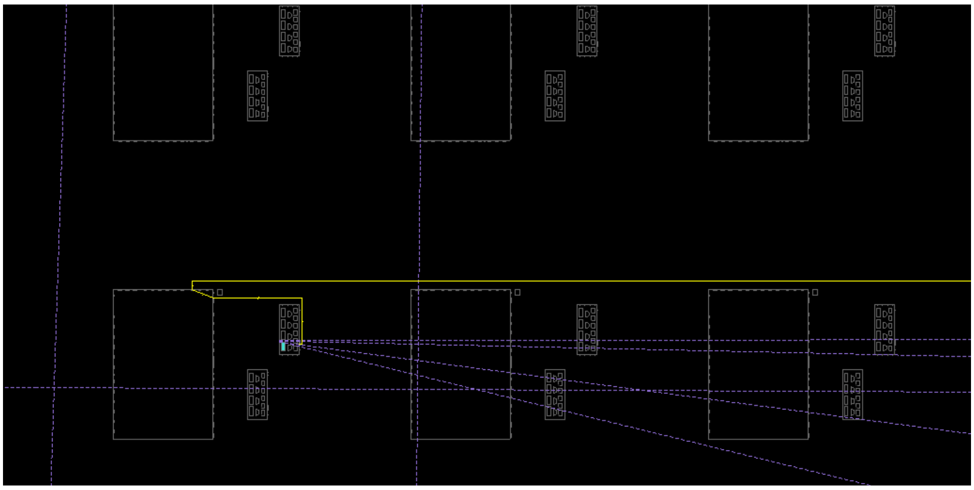


Figure 5-8 Manual wiring branch successful

If the selected pins cannot be routed successfully, a failure reminder will be given. As shown in Figure 5-9.

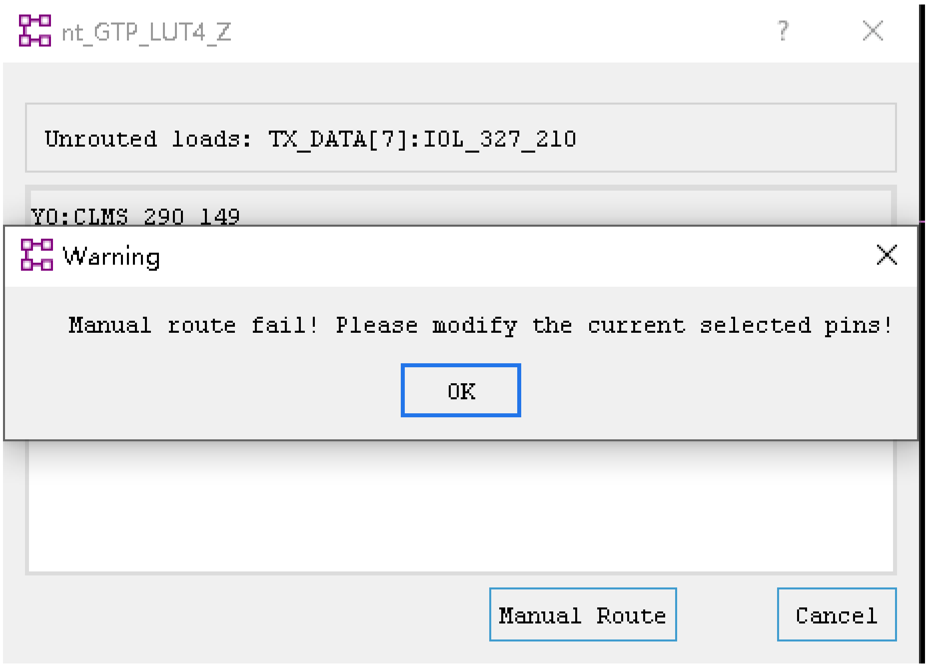


Figure 5-9 Wiring failure prompt

5.1.2 Disconnecting stitches

In the state of laying the line, click a branch with the mouse to select the branch and perform "Unroute", as shown in Figure 5-10.

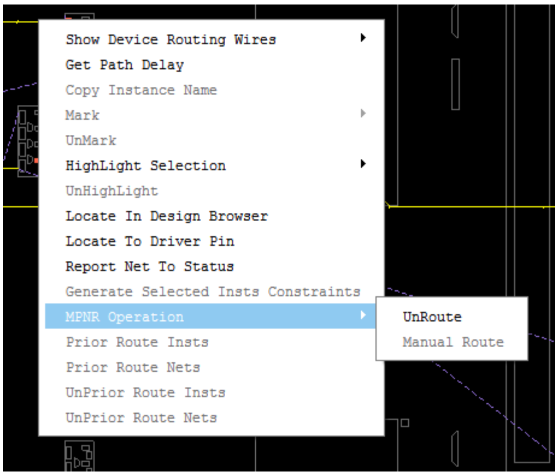


Figure 5-10 Select a branch to remove stitches

Figure 5-11 shows the effect after removing the connection of the branch.

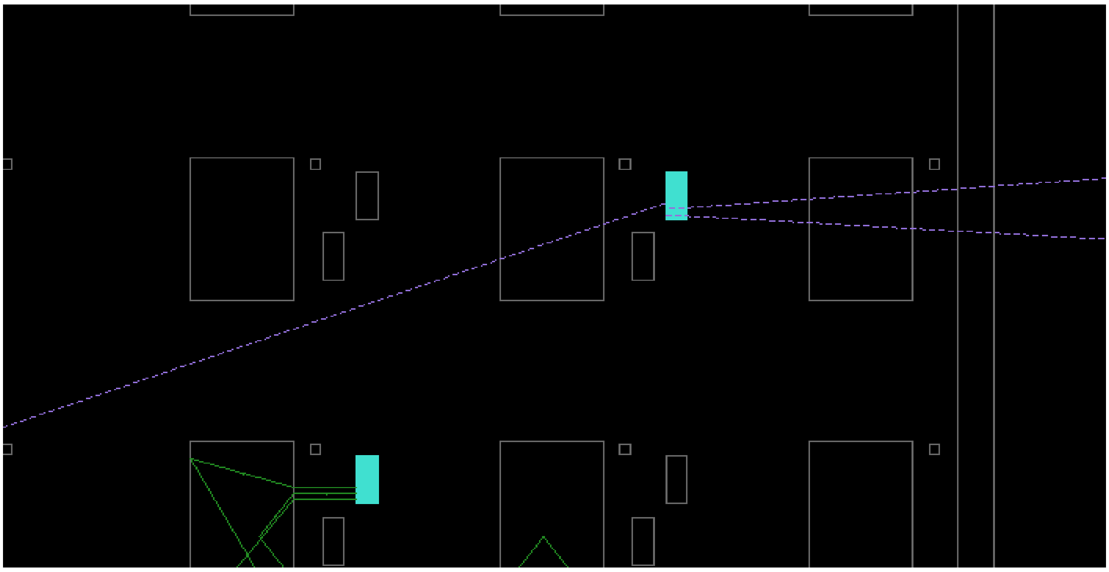


Figure 5-11 Deleting a certain branch of net

5.2 Assign Net Priority Routing

Select a net in the Device interface, right-click the menu and click Prior Route Nets, as shown in Figure 5-12:

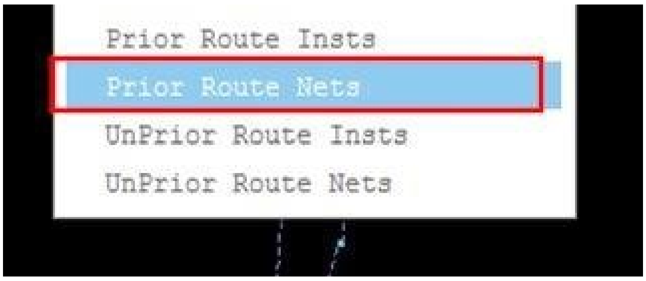


Figure 5-12 Prior Route Nets

You can mark the net, and the color of the marked net will change from purple to dark green.

5.3 Specify Inst Priority Routing

Select an already placed instace on the Device interface, and click its right-click menu to select Prior Route Insts, as shown in Figure 5-13.

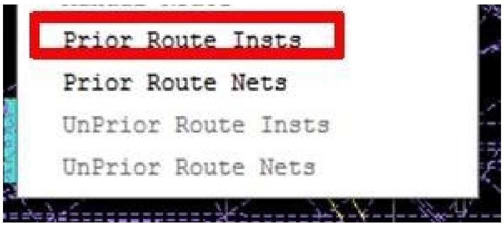


Figure 5-13 Prior Route Nets

You can mark the instance, and the color of the marked inst will change from turquoise to dark green.

5.4 Cancel net priority wiring

Select the marked Net, right-click the menu and click UnPrior Route Nets, as shown in Figure 5-14:

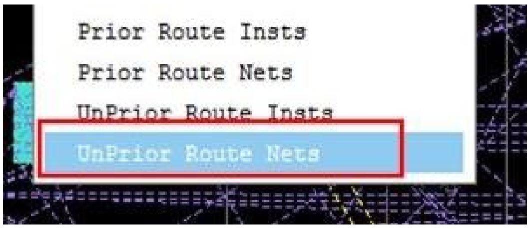


Figure 5-14 UnPrior Route Nets

The net will be unflagged, and the color of the unflagged net will change from dark green to purple.

5.5 Cancel Inst Priority Routing

Select the marked Inst, right-click the menu and click UnPrior Route Insts, as shown in Figure 5-15:

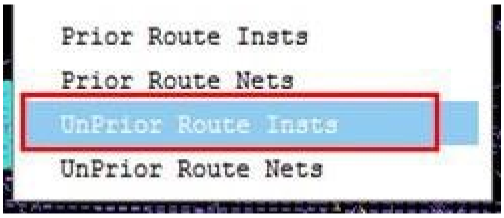


Figure 5-15 UnPrior Route Insts

The inst will be unmarked and the color of the inst will change from dark green to lime green.

5.6 File Constraints

Users can write their own Routing Constraint File (RCF) according to the format of routing constraints, and then then load the file when opening the RCE interface.

The RCF file supports line commenting of wiring constraints starting with #, but does not support block commenting of the form /**/.

5.7 Mark function

In the device interface, the instance resource can be marked with a "diamond" mark. At the same time, the mark browser synchronizes the mark information and distinguishes it by color. The mark browser can locate the location of a resource that is marked and remove the resource that does not need to be marked from the mark browser as follows.

1. For a selected resource, right-click and select Mark/Unmark, as shown in Figure 5-16.

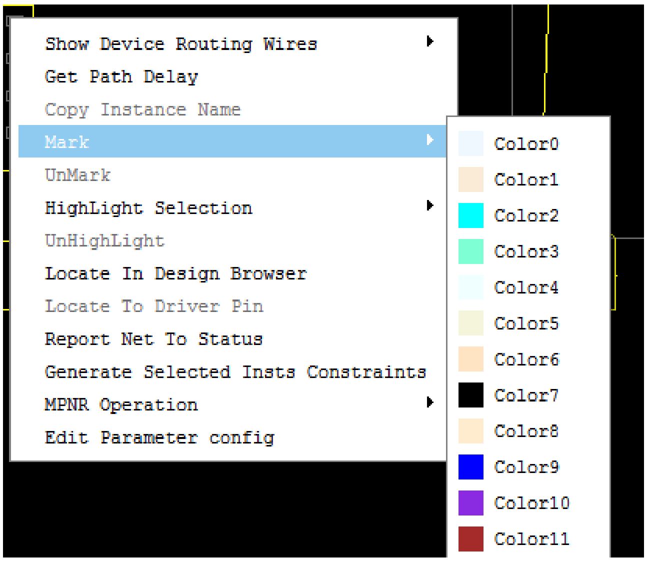


Figure 5-16 Setting up Mark

2. The effect of Mark resource is shown in Figure 5-17

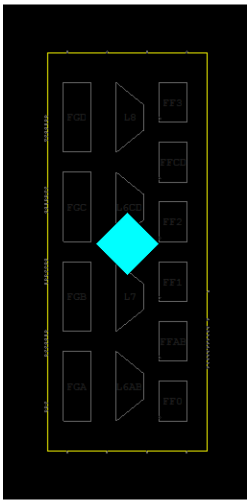


Figure 5-17 Device after setting Mark

3. In the toolbar, there is a button diamond to control whether the Mark browser window is displayed, as shown in Figure 5-18.

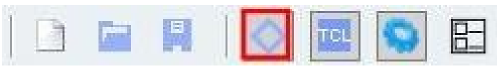


Figure 5-18 Mark browser button

4. The Mark browser is displayed as shown in Figure 5-19.

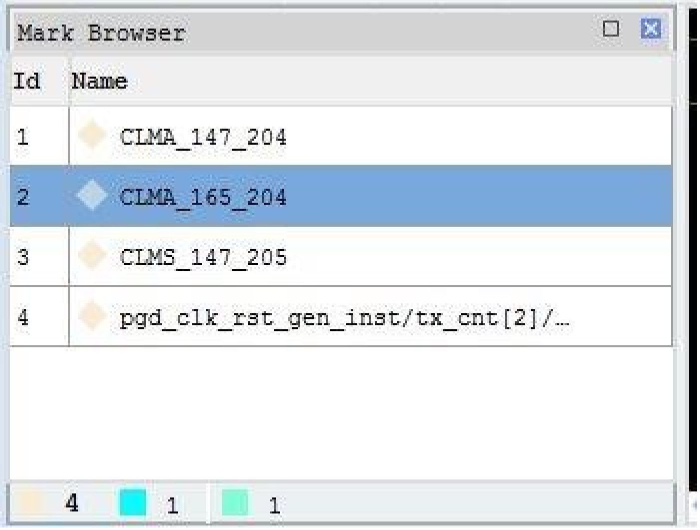


Figure 5-19 Mark browser window

5. You can also select the Locate function in the Inst Browser to jump to the corresponding inst and set mark and unmark for inst. You can also select the Locate function in the Inst Browser to jump to the corresponding inst, and set mark and unmark for inst.

6. In the same way, the Mark browser can remove unwanted resources by selecting Select the Unmark function in the right-click menu of the resource.

7. Currently, you can select inst or device in the device interface, inst browser to mark in the inst browser. If you select instanst in the inst browser, the same mark will appear on the corresponding location in the device interface. The same mark will appear on inst in the corresponding location on the device screen. If you select inst in any of the interfaces in mark browser, device interface, or inst browser to clear the mark, the mark of the other interfaces of the object will also be cleared.

5.8 Other functions

Press alt to open the zoom view in the device interface, hold down alt to drag the view in the device The zoom view allows normal view operations and The zoom view allows you to perform normal view operations and view related information, as shown in the following figure

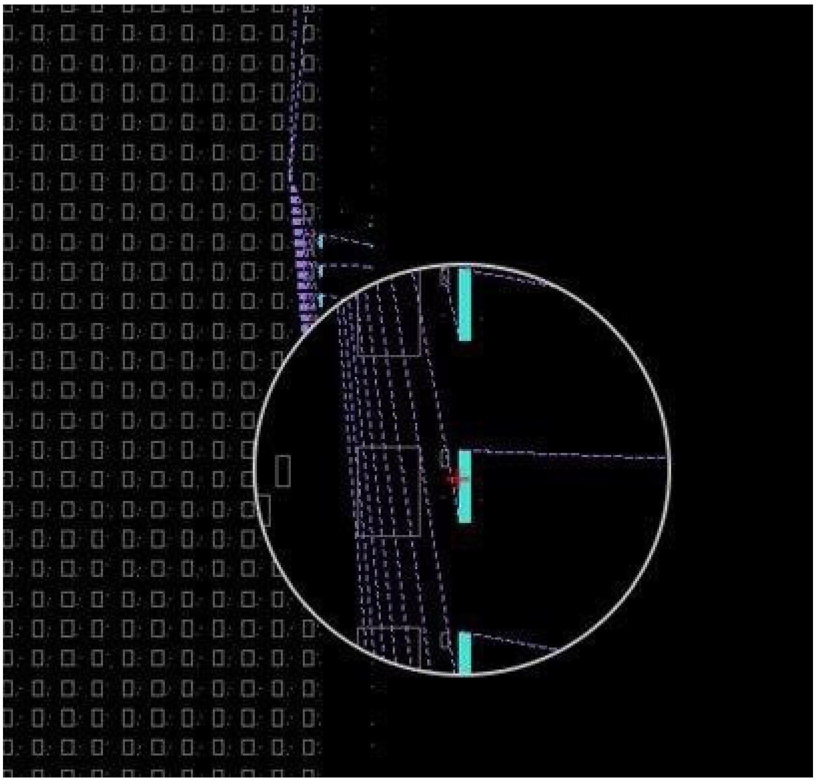


Figure 5-20 Local zoom view

The following pop-up window will appear when entering RCE or when the command is running, indicating that RCE is loading or or the command is running, and prevents the user from performing other interface operations, as shown in Figure:



Figure 5-21 Command Run Scrollbar

After the background button is clicked, the pop-up window disappears and the scroll bar appears in the status bar. The vpnr operation of the interface can be executed, and the vpnr command can be executed, but the mpnr command cannot be executed.

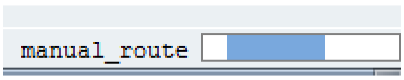


Figure 5-22 background scroll bar

5.9 Input commands

5.9.1 Close

The user can use this command to close the RCE interface.

5.9.2 get_path_delay

The user can use this command to calculate the path delay.

Command format: get_path_delay

get_path_delay

Parameter Explanation

-from is the startpoint of the path, the value can be a number of pin names or device names.

-though is the intermediate point that the path must pass through, the value can be several pin names or device though is the intermediate point through which the path must pass.

-to is the endpoint of the path, and can be the name of a pin or device.

-max_pax is the maximum path value, which can only be an integer.

-min specifies that the delay_type is min, and defaults to -max if not specified.

-fast specifies that corner is fast, or -slow if not specified.

-show_route prints out all nodes that the path passes through.

5.9.3 Help

You can use this command to print out all the commands supported by RCE.

5.9.4 manual_route

The user can use this command to perform manual routing operations on the specified design net branch. Use in the following environment.

Only one branch of a net can be specified for manual routing at a time, and that branch is in the unroute state.

The inst where the specified branch starts and ends must be constrained in the PCF file.

Command format.

manual_route -net {net} -branch {load} -pins {pin1;pin2;...pinx }

Parameter Explanation.

- net Specifies the design net object to which the manual routing branch belongs.

-branch Specifies which branch of the net to be manually wired.

-pins specifies a path between the start and end points, consisting of at least two device pins.

Example.

manual_route -net {AAL5_CRC_NEXT_114[12]} -branch {B4:CLMS_382_265}

-pins {IDE[1]:SRB_429_264>LH0:SRB_429_264; }

Branch B4:CLMS_382_265 on the net named AAL5_CRC_NEXT_114[12] (the branch is connected to port B4 of CLMS_382_265) is wired manually, and IDE[1]:SRB_429_264 to LH0:SRB_429_264 is a user-designed path, and the rest is done automatically by the system.

5.9.5 next

The user can use this command to display all the next available alternative pins for a given device pin. pins, which is required in the following environment.

Only a single device pin can be specified.

Command format: next

next -dev_pin:instance {pin:grid_device}

Parameter parsing: -dev_pin:instance

-dev_pin:instance specifies the current pin object and the grid_device where the current pin is located, thus uniquely identifies the pin.

Example.

next -dev_pin:instance {Q1:CLMA_117_115}

Show all the next available pins for the pin named Q1 on device name CLMA_117_115. All alternative pins are available.

5.9.6 save_adif

The changes made to the design in the RCE are saved to the current PLC DB file.

Parameter Explanation.

-file_name This parameter is currently ignored and the save as DB file function is not yet supported.

Example: -file_name

save_adif

5.9.7 select

The user can use the command to find and select the specified object.

Command format: select [-design_instance

select [-design_instance] [-design_net] [-dev_instance] [-nogroup]

Parameter Explanation.

-design_instance specifies the selected design instance, and if the design inst exists in a group, the other instances in the group are also selected

-dev_instance specifies the selected dev_instance and the design instance distributed on top of it

-nogroup This parameter should be used in conjunction with the previous parameter, if the previous parameter is -design_instance indicates that only the design instance is selected and not the other instances in its group; if the preceding argument is -dev_instance, then only the design instance is selected. parameter is -dev_instance, it means that if the design instance laid out on the specified device instance exists in a group, only the specified device instance will be selected and not the other instances in the group. If the preceding parameter is -dev_instance, it indicates that if the design instance on the specified device instance exists in a group, only the specified device instance is selected and not the device instance of the other instances in the group

-design_net specifies the selected design_net

Note: -design_net and -design_instance can be used together, but -design_net/inst cannot be used in conjunction with -dev_inst

Example.

select -design_net {net_01;net_02}

Select the design net named net_01 and net_02

select -design_instance {inst_01;inst_02}

Selects the design instances named inst_01 and inst_02 and all other design instances in their group All design instances in their group

select -design_inst {inst_01;inst_02} -nogroup

Select the design instances named inst_01 and inst_02 without checking all other design instances in their group all other design instances in their group

select -dev_instance { CLMA_X; CLMA_Y }

Select the device instances named CLMA_X and CLMA_Y and the other design instances in the group on which they are laid out design instance in the group on which it is laid out

select -dev_instance { CLMA_X; CLMA_Y } -nogroup

Selects the device instances named CLMA_X and CLMA_Y and the design instances laid out on top of them. design instance on which they are laid out, but not the other instances of the group in which these instances are located.

5.9.8 show

The user can use the command to show the wiring results of the specified design net and the layout results of the device instance.

The command displays the wiring results of the specified design net and the layout results of the device instance.

The command format is

show [-design_net] [-dev_instance]

Parameter Explanation

show [-design_net] [-dev_instance] Parameter Explanation: -design_net Displays the wiring result of the specified design net, and outputs all its branches and their pins.

-dev_instance Displays the layout result of the specified dev_instance, and outputs all the instances of the layout above it. instance information

Example.

show -design_net {net_01;net_02}

Output all branches and their pins through net_01 and net_02 in command console

show -dev_instance { CLMA_X; CLMA_Y }

Output all instances on CLMA_X and CLMA_Y to command console

5.9.9 unroute

This command allows the user to complete the unroute operation for the specified design net and branch.

To use it in the following environment.

The specified design net must be in the routed state.

One or more design nets or net branches can be specified.

Command format: unroute -design_net

unroute -design_net {net} -branch{ net:load:dev_instance }

Parameter Explanation.

-design_net Specifies the design net object to unroute, optionally, separated by a semicolon if there are more than one.

-branch Specifies the branch to be torn, in the format of net_name:load_name:dev_instance_name. Optional, separated by a semicolon if there are more than one.

Example.

auto_route -design_net {inst_361/inst_87/inst_45/count_temp_s [6]; } -branch {ad_read_tpram_c[1]:B1:CLMA_181_4; }

The design_net named inst_361/inst_87/inst_45/count_temp_s [6] and the design_net named ad_read_tpram_c [1] to a branch on the design net that is connected to port B1 on CLMA_181_4.

6. HOW TO MANUALLY SPECIFY THE WIRING PATH DIRECTLY USING THE RCF FILE

If you do not want to open the RCE interface to manually specify the wiring path, you can do so as follows (PGL50H is an example). However, it is recommended to use the RCE interface to specify the wiring path manually, as this will ensure the correctness of the wiring constraints saved in the rcf file, and the interface is easier and faster to operate.

The details of how to specify the wiring path manually using the rcf file are as follows.

1. First, run the project to Place&Route, open DE, and enter mpnr mode.

For example, create a new PGL50H project, load the CLM_LUT.v file, run to pnr and open DE.

2. Find the net name that you need to specify the wiring path in the Net Browser of the Design Browser of DE or Device interface.

If you want to specify the wiring path of the critical path, you need to determine the net name of the branch where the critical path is located.

For example, the selected net name is nt_GTP_LUT4_Z.

3. Execute the show -design_net net_name command in the Console window of DE to print all the devices and their pins through which the net passes.

Note: After -design_net, fill in the name of the design net to be printed, and the command will print all the devices and their pins that the net passes through. The command will print all the devices and their pins that the net passes through.

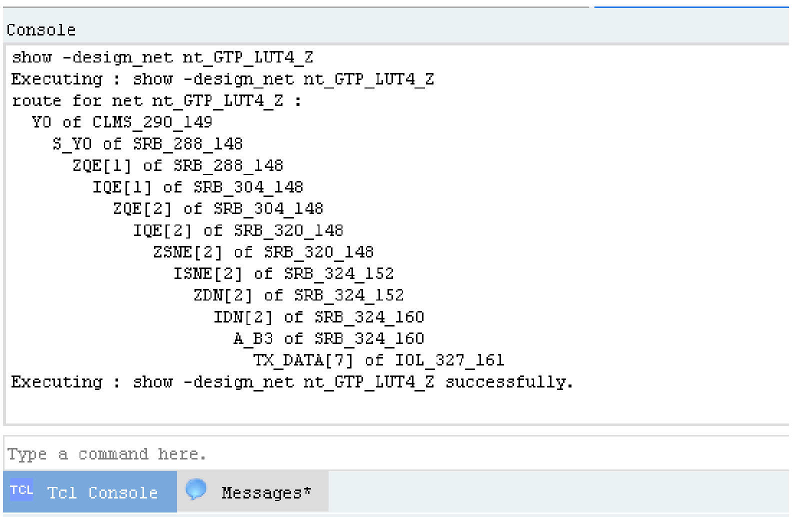


Figure 6-1 Show -design_net nt_GTP_LUT4_Z Execution Result

For example, if you execute the command show -design_net nt_GTP_LUT4_Z, the execution result is shown in the figure above.

4. Fill the wiring path information of net obtained in step 3 into the newly created rcf file, and add the rcf file to the tcl file using the add_constraint command (as shown in the tcl example at the end of the document).

The format of the constraints for specifying the wiring path in the rcf file is def_route {net_name}

{pin_name1:device_name1;pin_name2:device_name2;......;pin_nameN:device_nameN;}, noting that each pair of pin_name and device_name must correspond to each other and from the starting point (i.e. pin_name1:device_name1) to the end point (i.e. pin_nameN). The order of the complete connection from the starting point (i.e., pin_name1:device_name1) to the end point (i.e., pin_nameN:device_nameN) cannot be changed, i.e., the order of writing is the same as that of The order of the pins in the route of net nt_GTP_LUT4_Z obtained from step 3 is identical.

For example, create a new test.rcf file and fill in the route of net nt_GTP_LUT4_Z obtained from the Console in step 3.

Path information: def_route {nt_GTP_LUT4_Z

def_route {nt_GTP_LUT4_Z}

{Y0:CLMS_290_149;S_Y0:SRB_288_148;ZQE[1]:SRB_288_148;IQE[1]:SRB_304_148;ZQE[2]:

SRB_304_148;IQE[2]:SRB_320_148;ZSNE[2]:SRB_320_148;ISNE[2]:SRB_324_152;ZDN[2]:SRB_324_152;IDN[2]:SRB_324_160;ZQN[2]:SRB_324_160;
IQN[2]:SRB_324_176;ZQN[3]:SRB_324_176;IQN[3]:SRB_324_196;ZDN[3]:SRB_324_196;IDN[3]:SRB_324_204;ZSN[3]:SRB_324_204;ISN[3]:SRB_324_ 208;S_CD:SRB_324_208;TX_DATA[7]:IOL_327_210;}

5. In the Device interface of DE, find all the instances or ports corresponding to the starting and ending points of the net, and record the device names where these instances and ports are placed respectively.

If you only need to specify the wiring path of a critical path, you only need to find the starting and ending instances or ports of the branch where the critical path is located and their names.

If you only need to specify the cabling path of a critical path, you only need to find the start and end points of the branch where the critical path is located and the device name where it is placed. If you need to specify the wiring path of the entire net, you need to find the instance or port and the device name of the start and end points of all branches of the net.

For example, the driver pin of the above nt_GTP_LUT4_Z corresponds to the instance T_GTP_LUT4/gateop_perm. The device placed is FYA of CLMS_290_149; the load pin corresponds to GTP_LUT4_Z_obuf/opit_1, and the device placed is IOL_327. device is IOL_327_161.

6. Open the PCE, put the instance and port of the net or the start or end point of each branch found in step 5 into the corresponding device. The device is IOL_327_161. pcf file when running DeviceMap, i.e., do not check the Override .pcf option in the DeviceMap configuration options).

For example, in the PCE, add the instance (T_GTP_LUT4/gateop_perm) and port (GTP_LUT4_perm) found in step 5 to the PCE. (GTP_LUT4_Z_obuf/opit_1) found in step 5 are bound to FYA and P20 of CLMS_290_149 respectively. The saved pcf file is as follows

def_inst_site {T_GTP_LUT4/gateop} CLMS_290_149 FYA

def_port {GTP_LUT4_Z} LOC=P20 VCCIO=1.2 IOSTANDARD=LVCMOS12

SLEW=SLOW DRIVE=2 NONE=TRUE

7. If you re-run to pnr and open DE, you can see that the wiring path of nt_GTP_LUT4_Z is exactly the same as the one specified in the rcf file.

The wiring path of nt_GTP_LUT4_Z is exactly the same as that in the rcf file.

Note: The description of the tcl file mentioned in step 4.

(1) Note the order and location of the pcf and rcf files.

(2) The dev_map option needs to be override_pcf false, so that the pcf file is not overwritten.

(3) Example of tcl file is as follows (take PGL50H as an example)

set_arch -family Logos -device PGL50H -speedgrade -6 -package FBG484

add_design -verilog "CLM_LUT.v"

synthesize -ads -selected_syn_tool_opt 2

dev_map -override_pcf false

add_constraint "1.pcf"

add_constraint "1.rcf"

pnr

report_timing

gen_bit_stream}

7. CONTACTS

Контакты для технических и коммерческих вопросов
ООО «Инэк»
г. Санкт-Петербург, ул. Яблочкова, д. 20, литер А, оф. 504
contact@innek.ru
+7 (812) 200-40-37